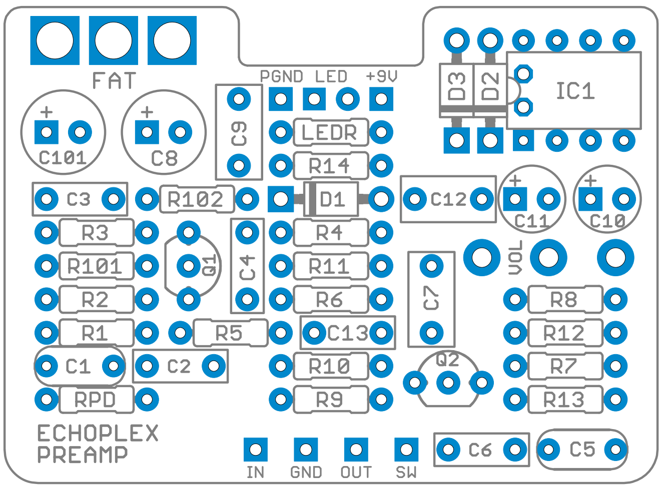
Ares Preamp

Maestro Echoplex EP-3 Preamp

aion electronics

Overview

Ares Project Link



The Echoplex EP-3 tape echo unit was originally released in 1970. It became legendary not only for its tape delay effect, but also for the coloration that its preamp would add even with the echo turned off. It's a pretty simple FET boost, but it does something special with the phase response of the signal.

The Ares Preamp is a reproduction of the EP-3 unit in bypass mode. It uses a charge pump to get 18 or 24v from a 9 or 12 volt source. There is also an output buffer on the end (adapted from the ClinchFX EP-Pre) that preserves the tone but gives it a much lower output impedance than the original.

There were two versions of the EP-3, the "early version" which was brighter and the "late version" which was a little darker and fatter. The Ares has a switch that allows you to switch between these two versions, as well as adding in another capacitor that thickens the tone even more.

Controls & Usage

The Echoplex preamp controls are pretty simple:

- **Volume** starts at zero and tops out just slightly above unity with the Fat switch in the center position. The maximum volume is around 2:00 but there is still a slight tonal change from 2:00 to 5:00. The boost level is increased with the other two switch positions.
- **Fat** lets you go from early to late model EP3 as well as adding a third option to thicken the tone even further. The middle switch position is the early EP3 circuit, which many people prefer.

Selecting FETs

The TIS58 FETs used in the original EP-3 are nearly impossible to find. Even if you did find some they won't help you too much: FET manufacturing is notoriously inaccurate and you'll find an enormous variance even within one part number. Not surprisingly, some original EP-3 units just didn't have the "it" factor. The preamp is such a simple circuit that the FET really does make it or break it.

The best way to ensure that the FETs give the proper response is to test their I_{DSS} value, which can be done with the circuit described here. I measured the FETs in an original ClinchFX EP-Pre. He sorts his FETs and only uses ones that fall inside certain parameters. I would recommend using his values as targets when selecting your own FETs. Here's what I found:

Using a 9.6V wall wart supply and the circuit here, Q1 (the 2N5484) had an I_{DSS} of **2.89mA** while the Vp measured **-1.32V**. Q2 (the 2N5485) had an I_{DSS} of **7.62mA** and a Vp of **-2.70V**. Fortunately, these values fall right about in the middle of the specified ranges for both parts, meaning it should be easy to find ones that will work. I would order maybe five of each of them and select the one that most closely matches the above values. You can even use other FETs as long as they measure the correct value—in my test build I used a 2N5952 for Q1 and it sounds identical to a 2N5484 with the same measurements.

Parts

Resistors

R1	100k
R2	1M
R3	3k3
R4	22k
R5	220k
R6	100k
R7	470k
R8	220k
R9	1M
R10	10k
R11	15k
R12	22k
R13	100k
R14	60R [1]
R101	1M [2]
R102	1M [2]
RPD	1M to 2M2 [3]
LEDR	4k7

Capacitors

C1	220pF MLCC
C2	22n
C3	22n
C4	100n
C5	220pF MLCC
C6	22n
C7	470n
C8	47uF electro
C9	470n MLCC [4]
C10	10uF electro
C11	10uF electro
C12	470n MLCC [4]
C13	22n
C101	47uF electro

Semiconductors

Q1	2N5484 [5]
Q2	2N5485 [5]
IC1	LT1054CP [6]
D1	1N4743 zener [7]
D2, D3	1N5819 [8]
LED	5MM

Potentiometer

Volume	500kB

Switch

Fat	SPDT center off

[1] **Close enough:** Anything from 47R to 100R would be fine here.

[2] **Optional:** These 1M resistors are here to keep the Fat switch from popping when switched. They can be omitted if you don't care about this. (Just leave them off entirely—do not jumper them.)

[3] **The ClinchFX version** uses a 4M7 resistor for the pulldown value. Shouldn't make a difference.

[4] **Capacitor type:** I recommend using X7R MLCCs here, but film capacitors will work fine as well.

[5] **Selected FETs:** Q1 should have an IDSS value of around **3mA**, right in the middle of the range of the 2N5484. Q2 should have an IDSS value **between 7mA and 8mA**. The part numbers don't have to be the ones specified as long as they have those measurements. See previous page for more information.

[6] **Charge pump:** An LT1054 is recommended for the charge pump, but you can also use a TC1044SCPA. If using a TC1044, jumper the "OSC" pads under the IC socket or you will hear an audible whine.

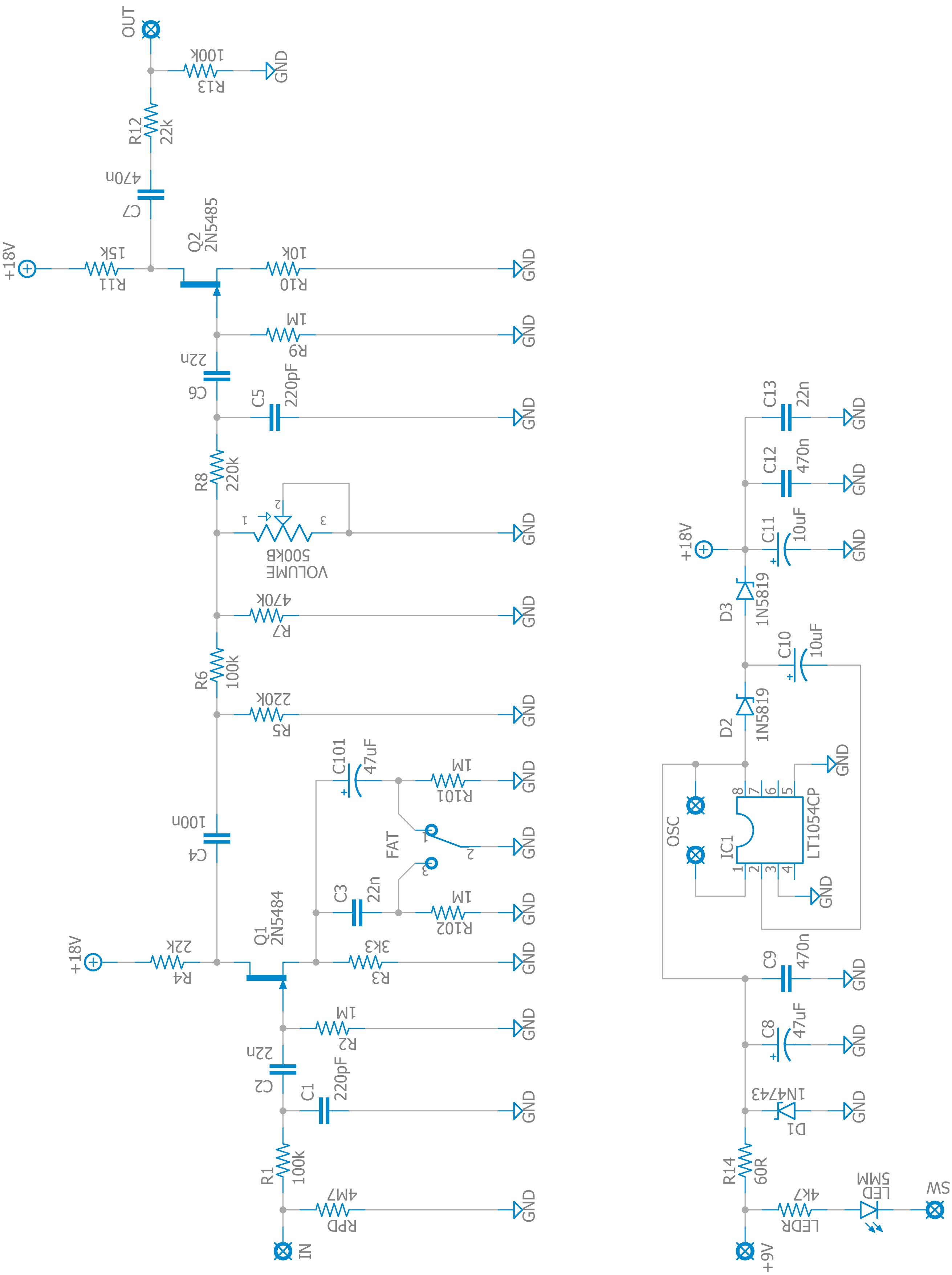
[7] **Zener diode:** The LT1054 max input voltage is 15V, and the ClinchFX unit uses a 1N4743 which is a 13V zener. You can sub a more common 1N4742 (12V zener) as well. This is required if using a TC1044.

[8] **Charge pump diodes:** The 1N5819 has a very low voltage drop which helps keep the voltage as high as possible. A 1N400(X) diode can be used as well—just know that the resulting voltage will be about 1V less.

Additional Part Notes

- Capacitors are shown in nanofarads (n or nF) where appropriate. 1000n = 1uF. .
- The PCB layout assumes the use of film capacitors with 5mm lead spacing for all values 1nF through 470nF. I prefer EPCOS box film or Panasonic ECQ-B/V-series.
- Potentiometers are Alpha 16mm right-angle PCB mount.
- I recommend using these dust covers / insulators from Small Bear to insulate the back of the pots.

Schematic


OUT
R13
100K
GND
R12
22k
C7
470n
Q2
2N5485
+18V
R11
15K
R10
10K
R9
1M
C6
22n
C5
220pF
R8
220k
VOLUME
500KB
R7
470K
R6
100k
R5
220K
C101
47uF
R101
1M
FAT
C4
100n
C3
22n
R102
1M
Q1
2N5484
+18V
R4
22K
R3
3K3
R2
1M
C2
22n
C1
220pF
R1
100k
RPD
4M7
IN
C13
22n
C12
470n
+18V
C11
10uF
D3
1N5819
C10
10uF
D2
1N5819
OSC
IC1
LT1054CP
C9
470n
C8
47uF
D1
1N4743
R14
60R
LEDR
4K7
LED
5MM
SW
+9V

General Build Instructions

These are general guidelines and explanations for all Aion Electronics DIY projects, so be aware that not everything described below may apply to this particular project.

Build Order

When putting together the PCB, it's recommended that you do not yet solder any of the enclosure-mounted control components (pots and switches) to the board. Instead, follow this build order:

1. Attach the **audio jacks**, **DC jack** and **footswitch** to the enclosure.
2. Firmly attach the **pots** and **switches** to the enclosure, taking care that they are aligned and straight.
3. Push the **LED**[1] into the hole in the enclosure with the leads sticking straight up, ensuring that the flat side is oriented according to the silkscreen on the PCB.
4. Fit the **PCB** onto all the control components, including the leads of the LED. If it doesn't fit, or if you need to bend things more than you think you should, double-check the alignment of the pots and switches.
5. Once you feel good about everything, **solder them from the top**[2] as the last step before wiring. This way there is no stress on the solder joints from slight misalignments that do not fit the drilled holes. You can still take it out easily if the build needs to be debugged, but now the PCB is "custom-fit" to that particular enclosure.
6. Wire everything according to the wiring diagram on the last page.

[1] **For the LED**: You can use a bezel if you'd like, but generally it's easier just to drill the proper size of hole and push the LED through so it fits snugly. If you solder it directly to the PCB, it'll stay put even if the hole is slightly too big. Make absolutely sure the LED is oriented correctly (the flat side matches the silk screen) before soldering, as it'll be a pain to fix later! After it's soldered, clip off the excess length of the leads.

[2] **Note on soldering the toggle switch(es):** It will require a good amount of solder to fill the pads. Try to be as quick as possible to avoid melting the lugs, and be prepared to feed a lot of solder as soon as the solder starts to melt. I recommend waiting 20-30 seconds between soldering each lug to give it time to cool down.

"RPD" and "LEDR" resistors

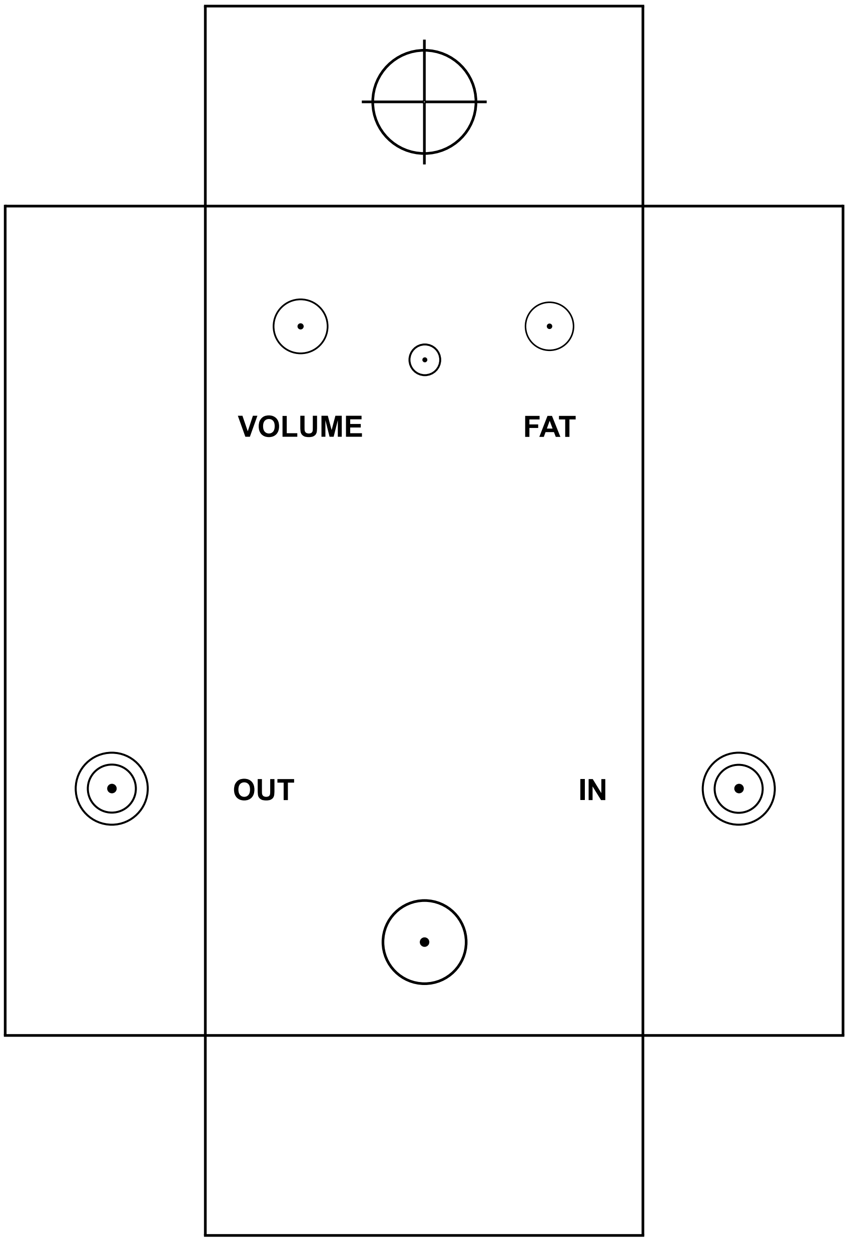
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

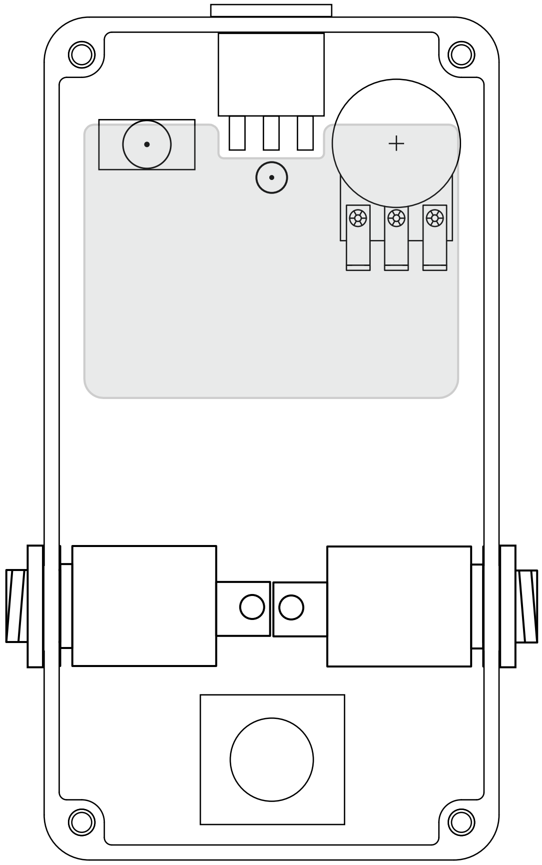
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and cut out the drilling template below. Tape it to the enclosure to secure it while drilling. Note that the holes are shown slightly smaller than they need to be, so drill out the holes as shown and then step up until they are the correct size for the components.



Hammond 1590B
(bottom/inside view)



Parts Used

- Switchcraft 111X enclosed jacks
- Kobiconn-style DC jack with internal nut

Standard Wiring Diagram

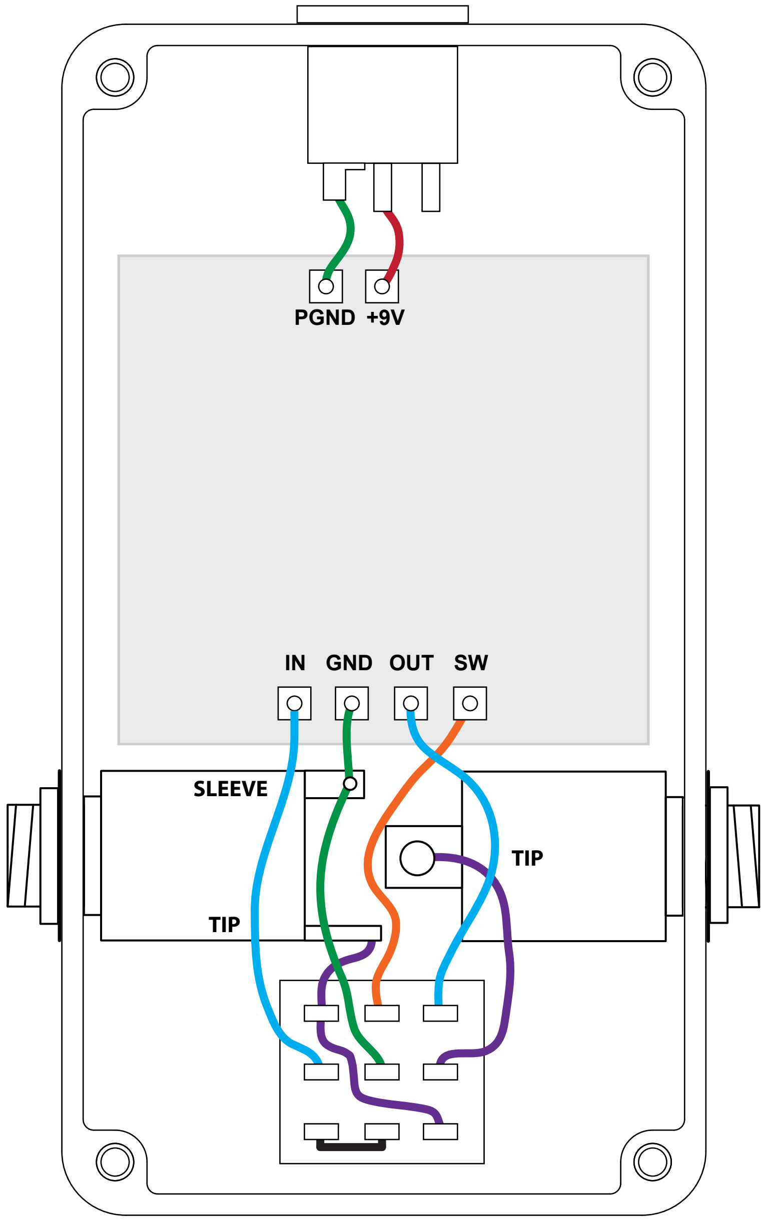
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

The **SW** pad is the cathode connection for the LED. This will connect to ground to turn it on when the switch is on. Usage of the on-board LED connection is not required if you have specific placement needs for your enclosure, but's incredibly convenient.

The wiring diagram also makes use of **star grounding** principles where all of the grounds connect to a single ground point (in this case the sleeve of the input jack). This is best practice to avoid added noise caused by improper grounding. The sleeve of the output jack is unconnected.

If using a painted or powdercoated enclosure, **make sure both jacks have solid contact with bare aluminum** for grounding purposes. You may need to sand off some of the paint or powdercoat on the inside in order to make this happen.

Make sure to double-check the markings of the pads on the PCB for your particular project – they are not always in the order shown here!



License / Usage

No direct support is offered for these PCBs beyond the provided documentation. It is assumed that you have at least some experience building pedals before starting one of these. Replacements and refunds will not be offered unless it can be shown that the circuit or documentation are in error. I have in good faith tested all of these circuits. However, I have not necessarily tested every listed modification or variation. These are offered only as suggestions based on the experience and opinions of others.

Projects may be used for commercial endeavors in any quantity unless specifically noted. No bulk pricing or discounting is offered. No attribution is necessary, though a link back is always greatly appreciated. The only usage restrictions are that **(1) you cannot resell the PCB as part of a kit**, and **(2) you cannot "goop" the circuit, scratch off the screenprint, or otherwise obfuscate the circuit to disguise its source**. (In other words: you don't have to go out of your way to advertise the fact that you use these PCBs, but please don't go out of your way to hide it. The guitar effects pedal industry needs more transparency, not less!)